



Fluid Dynamics, Power Generation and Environment Department Single Phase Thermal-Hydraulics Group

6, quai Watier F-78401 Chatou Cedex

MARCH 2024

code_saturne documentation

code_saturne version 8.0 tutorial: three 2D disks

 $contact: \ saturne-support@edf.fr$



TABLE OF CONTENTS

	I Introduction	5
1	Introduction	6
1.1	CODE_SATURNE SHORT PRESENTATION	6
1.2	About this document	6
1.3	CODE_SATURNE COPYRIGHT INFORMATIONS	6

7

15

II Three 2D disks

1 Study description 8 1.18 STUDY CREATION AND PREPARATION 1.28 1.3Geometry 9 1.4 9 1.5DATA SETTINGS FOR SOLID AND FLUID DOMAINS 11 2 Computations of the 3 2D disks configuration 122.12.22.3Results for the uncoupled cases 12 2.4132.5

III Step by step description

1	Detailed tutorial step by step	16
1.1	Creation of the study in a terminal	16
1.2	LAUNCHING SYRTHES	17
1.3	Home Tab	18
1.4	File Names Tab	18
1.5	Conduction Tab	21
1.6	Control Tab	23
1.7	Output	24
1.8	RUNNING OPTIONS	25

1.9	PREPARING AND LAUNCHING THE CODE_SATURNE COMPUTATION ALONE	26
1.10	Mesh Tab	27
1.11	CALCULATION FEATURES TAB	30
1.12	VOLUME CONDITIONS	32
1.13	Boundary conditions	34
1.14	TIME SETTINGS TAB	35
1.15	Postprocessing Tab	35
1.16	RUN COMPUTATION	36
1.17	Postprocessing Analyses	36
1.18	PREPARING AND LAUNCHING CODE_SATURNE-SYRTHES COUPLED COMPUTATION	37
1.19	COUPLING SYRTHES- SYRTHES COUPLING	37
1.20	Coupling SYRTHES- Conjugate heat transfer Tab	38
1.21	Coupling SYRTHES- Control Tab	38
1.22	Coupling code_saturne- Boundary conditions Tab	39
1.23	Coupling code_saturne- Coupling parameters Tab	39
1.24	Coupling code_saturne- Time settings Tab	40
1.25	Coupling code_saturne- Numerical Parameters Tab	40
1.26	COUPLING RUNNING COMPUTATION	41

R&D
TOOL

Part I

Introduction

EDF R&D

1 Introduction

1.1 code_saturne short presentation

code_saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D axisymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. code_saturne relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes).

1.2 About this document

The present document is a tutorial for code_saturne version 8.0. It presents test cases and guides the future code_saturne user step by step into the preparation and the computation of the cases. It focuses on code_saturne SYRTHES coupling feature allowing to run simulations taking into account conjugate heat transfers.

The test case directories, containing the necessary meshes and data are available in the examples/4-2Ddisks directory in code_saturne source directory.

This tutorial focuses on the procedure and the preparation of the code_saturne and SYRTHES computations with or without SALOME. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

The first part is this introduction. In the second part, the configuration, geometry, data settings, numerical parameters are described for a solid computation alone, a fluid computation alone and finally for a coupled fluid/solid computation. In the third part, a detailed description of the steps to be followed is given.

1.3 code_saturne copyright informations

code_saturne is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. code_saturne is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

Part II

Three 2D disks

1 Study description

1.1 Objective

This tutorial case focuses on code_saturne coupling feature with SYRTHES, an open source code solving thermal conduction and radiative transfer problems in solids, which is one of the way to model conjugate transfer phenomenon with code_saturne.

The aim is to simulate the natural convection flow of air inside a sheath containing three electric wires. A 2D cross section of the sheath is considered. Thermal conduction inside the three wires is solved with SYRTHES and coupled with the flow solved with code_saturne.

In a first step, uncoupled solid and fluid calculations will be set and run separately, and only in a second step, a coupled solid/fluid calculation. The data settings used for the uncoupled step can largely serve as basis for the coupled step.

1.2 Study creation and preparation

First, create the \bigcirc 3_2D_DISKS study directory, with two subdirectories \bigcirc FLUID and \bigcirc SOLID. Two ways are available:

• Create coupled study with SALOME module CFDSTUDY as described on Figure II.1.



Figure II.1: Coupled study creation with SALOME module CFDSTUDY.

At this step, it is possible to choose the total number of processes that will be used for a coupled computation. Here 2 is chosen.

Create coupled study in a terminal with the following command line:
 \$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID

Second, mesh files have to be copied in the study. The fluid mesh has to be copied in the subdirectory \bigcirc MESH. The solid mesh must be copied in the subdirectory \bigcirc SOLID.

Third, to set up the solid settings, launch SYRTHES Graphical User Interface (GUI) in SALOME by

		code_saturne
EDF R&D	code_saturne version 8.0 tutorial: three 2D disks	documentation Page 9/41

opening SYRTHES module¹ or in a terminal by typing **\$ syrthes.gui &** inside the subdirectory **SOLID**.

Fourth, to set up the fluid settings, launch the code_saturne GUI in SALOME or in a terminal inside the subdirectory FLUID/DATA (script SaturneGUI) for the fluid computation alone.

1.3 Geometry

As said above, the simplified configuration represents a 2D cross section of the electric wires inside the sheath. The 3 wires inside the sheath are represented as 3 disks inside a larger ring and it is assumed that the 3 disks are in contact with an air flow inside the electric sheath.

The geometry is shown on figure II.2.



Figure II.2: Geometry with the solid domain (parts 1, 2, 3, 4) and the fluid domain (part 5).

1.4 Meshes characteristics

• <u>Mesh of the solid domain</u>:

The mesh of the solid part contains 11688 nodes (P_1 discretization) and 5688 elements. Materials properties and boundary conditions are specified in the next part using the references given on figure II.3.

Type: unstructured mesh Mesh generator used: SIMAIL

• <u>Mesh of the fluid domain</u>:

The fluid mesh contains 3866 nodes. Mesh Quality Criteria run type in the code_saturne Graphical User Interface (Calculation management, Pepare batch calculation section) can be used to check the quality of the mesh and to help to identify the references associated to the boundary conditions (1 is used for all solid boundaries, 2 for the front fluid face, and 3 for the back fluid face). See figure II.4.

Type: unstructured mesh Mesh generator used: SIMAIL

 $^{^{1}}$ Once a .syd file is present, you can right-click on it in CFDSTUDY object browser and launch SYRTHES GUI from the contextual menu.



Figure II.3: References for the solid interior zones (black) and solid boundary zones (pink).



Figure II.4: fluid boundary faces references

1.5 Data settings for solid and fluid domains

• Solid domain:

Material properties for the solid domain are given by part (1 to 3 for the electric wires and 4 for the disk for the electric sheath) in the subsequent table:

	Conducti	vity type	Values	$(W/m/^{\circ}C)$		Volume reference
Disk 1	Isotropic		$k_{11} = 25$			1
Disk 2	Orthotropi	с	$k_{11} = 25$; $k_{22} = 5$		2
Disk 3	Anisotropie	C	$k_{11} = 25$; $k_{22} = 5$	$\alpha = 45^o$	3
Disk 4	Isotropic		$k_{11} = 25$			4
		Physical	properti	es Values		
		Density $[\rho$)	7700	(kg/m^2)	
		Specific he	eat $[C_p]$	460	$(J/kg/m^3)$	3)

Initial and boundary conditions for the solid domain should be defined as follows:

Initial conditions	
Temperature condition	$T_{ini,s} = 20^{\circ} \mathrm{C}$

Boundary conditions	Value	Surface reference
Heat exchange conditions $(q_{w,ext})$	$T_{ext} = 90^{\circ}$ C.; $h_{ext} = 1000(W/m^2.K)$	2 or 5 or 8

For the coupled case, nothing needs to be changed in this part.

• Fluid domain:

Some characteristics of the air flow that should be checked in the GUI are given hereafter:

Modeling feature	choice		
Time step	constant in time and uniform in space		
Turbulence model	$k - \varepsilon$ Linear Production		
Thermal model	Temperature (°C)		

In the air flow, density is a function of the temperature and gravity force is taken into account. The 3 disks, which are warmer than the air flow, generate a temperature difference creating a fluid movement. The warmer air flow is moving to the top and the colder air flow to the bottom of the fluid domain.

The density follows an ideal gas law that can be specified in the GUI:

$$\rho = \frac{p_0}{R_g \ (T+273.15)} \tag{II.1}$$

where ρ is the density, T is the temperature (°C), $R_g = 287 \ (m^2 \cdot s^{-2} \cdot K^{-1})$ the ideal gas constant and $p_0 = 101325 \ (Pa)$ the reference pressure (atmospheric pressure).

Initial conditions are defined below:

Initial conditions	
Temperature condition	$T_{ini,f} = 20^{\circ} \text{C}.$

Symmetry conditions are imposed on the front and back face, and walls are set on all remaining faces (those coupling fluid and solid domain).

Boundary conditions	Values	Surface reference
Walls (Heat exchange $q_{w,ext}$)	$T_{ext}(^{\circ}\mathrm{C})$; $h_{ext}(W/m^2.K)$	1
Symmetry		2 or 3

For the fluid computation alone $T_{ext} = 30^{\circ}$ C and $h_{ext} = 10W/m^2$. K while for the coupled case, simply set $T_{ext} = 0^{\circ}$ C and $h_{ext} = 0W/m^2$. K.

2 Computations of the 3 2D disks configuration

If not already done, set all data necessary to run the fluid and solid part configurations separately based on the description of the previous section. All the parameters necessary to this study can be defined through code_saturne GUI and SYRTHES GUI respectively.

2.1 Time stepping parameters

Set now all the time stepping parameters on both sides: code_saturne and SYRTHES.

Time stepping parameters of solid computation						
Reference time step	10 (s)					
Number of iterations	100					
Time stepping par	ameters of fluid computation					
Time stepping par Reference time step	ameters of fluid computation 0.1 (s)					

These time stepping parameters will be set to run the fluid and solid computations **independently** from one another.

2.2 Output management

Standard options for output management will be used. Only one monitoring point will be created for the solid conduction computation at the following coordinates:

Probe	x (m)	y (m)
1	0.003	-1.2

The output frequency for this probe can be every 10 time steps.

The temperature field can be saved every 25 time steps.

2.3 Results for the uncoupled cases

The velocity and **temperature** field can be post-processed with ParaVis.

As examples, Figure II.5 shows the evolution of the temperature in the solid domain and Figure II.6 the evolution of the temperature in the fluid domain without **Conjugate heat transfer**.



Figure II.5: The temperature evolution in the solid domain without coupling method



Figure II.6: The temperature evolution in the fluid domain without coupling method

2.4 Parameters for the coupled computation

Conjugate heat transfer has to be enabled and the coupling interfaces specified on both sides.

Then time stepping parameters have to be modified to be able to see the effect of the conjugate heat transfer phenomenon between the solid and fluid domains. For this reason, we increase the number of iterations and the reference time step for the fluid and the solid part².

Time stepping parameters of solid computation						
Reference time step	0.5~(s)					
Number of iterations	600					
Time stepping par	ameters of fluid computation					
Time stepping para Reference time step	ameters of fluid computation 0.5 (s)					

 2 By default, the smallest number of iterations will be used to drive the coupling computation. If we choose a number of iterations of 10000 for the fluid domain and 5000 for the solid domain, the coupling computation will be stopped after 5000 instead of 10000.



Figure II.7: Evolution of temperature and velocity magnitude

The **Improved pressure interpolation in stratified flow** option has to be checked in code_saturne GUI (Numerical parameters, Global parameters section).

2.5 Results for the coupled computation

As an example, Figure II.7 shows the evolution of the temperature in the solid and fluid area with the **conjugate heat transfer** enabled. The natural convection in the fluid domain due to the temperature difference imposed by the solid disks is clearly visible with the velocity displayed as a vector field.

Part III

Step by step description

1 Detailed tutorial step by step

1.1 Creation of the study in a terminal

• Step 1: check the post-install required for coupling code_saturne with SYRTHES. The first step is to check the post-install required for coupling with SYRTHES and verify if the SYRTHES •PATH is correctly known in the system environment. We just need to edit the batch file¹ name code_saturne.cfg as below:

\$ vim <install-prefix>/etc/code_saturne.cfg
>### Set the location to the SYRTHES installation directory.
> syrthes = <install-prefix-syrthes>

• Step 2: source the syrthes.profile file in your user environment. Before using SYRTHES alone, you have to copy and source this file to define SYRTHES environment

variables (like **\$SYRTHES4_HOME**) in your terminal, as follows:

```
$ cp <install-prefix-syrthes>/bin/syrthes.profile .
$ source syrthes.profile
```

\$ echo \$SYRTHES4_HOME (to check the SYRTHES PATH in your environment)

After having defined correctly your environment, to be able to launch a coupling computation code_saturne-SYRTHES or a SYRTHES computation alone, you just have to create the coupling study directory.

• Step 3: create the \bigcirc 3_2D_DISKS study directory, and the two case subdirectories \bigcirc FLUID and \bigcirc SOLID.

This is done using the standard command:

• **Remark**: The fluid mesh must be copied in the directory \bigcirc MESH. The solid mesh must be copied in the subdirectory \bigcirc SOLID.

¹see the installation guide, name install.pdf, in <install-prefix>/share/doc/code_saturne/ directory.

1.2 Launching SYRTHES

Preparation of SYRTHES computation alone can be one in 6 steps:

- Step 1: Launch SYRTHES module in SALOME or SYRTHES GUI in a terminal,
- Step 2: Create a New Data File,
- Step 3: Check the name of the mesh and convert this one in .syr format,
- Step 4: Define the initial and boundary conditions for the conduction problem,
- Step 5: Define the physical properties of each disk {1, 2, 3 and 4},
- Step 6: Running the SYRTHES computation alone.

• **Step 1**: launch SYRTHES module in SALOME or SYRTHES GUI in a terminal. SYRTHES GUI can be launched by the following command lines in the solid subdirectory:

\$ cd 3_2D_DISKS/SOLID/

\$ syrthes.gui &

• Step 2: choose New Data File inside the pop-up window.

Welcome to SYRTHES 5.0 GUI 5.0 @Copyright EDF 2011	
syrthes	
Create New Case New Data File Open Data F	ile

Figure III.1: Opening pop-up window for SYRTHES GUI in SALOME

1.3 Home Tab

Once New Data File is selected the Home Tab SYRTHES GUI appears.

'	
📄 🖆 🦺 🖏 Run SYRTHES 🕑 Stop SYRTHES 🚳 Calculation Prog	gress 📈
Home File Names Conduction User C functions Control Output Running options Dimension of the problem : 2D_cart • Additional physical modelling Thermal radiation Humidity Conjugate Heat Transfer SYRTHES 0D fluid flow	25

Figure III.2: Synthes Home Tab

1.4 File Names Tab

• Step 3: Select the mesh

Add the solid mesh in **Conduction Mesh** field. Choose the 2D solid mesh file with the format .des.

EDF R&D	code_saturne version 8.0 tutoria three 2D disks	al: code_saturne documentation Page 19/41
File Tools Prefere	ices Heln	
	Run SYRTHES 🕑 Sto	op SYRTHES 🔕 Calculation Progress 🗾
Home File Names Conduction Conjugate heat t User C functions Control Output Running options	Conduction input file name and location Conduction mesh: Radiation mesh: SYRTHES 1D fluid mesh: Restart File : Weather data (optional) : Conduction output files names prefix and location Results names prefix : resul	

Figure III.3: File Names Tab - Conduction Mesh

The SYRTHES (GUI) directly converts the .des to the .syr format.



Figure III.4: Converting mesh format

• Note: Inside SYRTHES GUI, we can load the SIMAIL format *.des for the solid mesh. This one will be automatically transformed to the *.syr format.

It can also be done with the following command line:

\$ convert2syrthes4 -m 3rond2d.des

• Remark: You can convert the *.syr format into a *.med format. Like that, you can load the *.med file inside SALOME, after having used this command line below:

\$ syrthes4med30 -m 3rond2d.syr -o 3rond2d.med



Figure III.5: Choose a name for the results files .res, .his and .rdt

EDF R&D

1.5 Conduction Tab

• Step 4: Define initial and boundary conditions

Initial conditions Define the initial temperature conditions inside the different disks. The reference value '-1' means 'applied for the whole domain'.

<u>F</u> ile <u>T</u> ools Preferences <u>H</u> elp						
📑 🚰 🏝 🦫 💐		Rur	SYRTHES	▶ Stop SYRTHES	Calculation Progress	\mathcal{M}
 Image: Conduction Service Conduction Imitial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options 	Initial temperature Type V Constant V Constant	Rur Clog C) Temperature 20 Image: Clog Comparison of Cl	References -1	Stop SYRTHES User 20°C everywhere	Calculation Progress	
	Constant +	-			[]	•

Figure III.6: Initial temperature conditions

Boundary conditions Define the temperature boundary conditions for the extern faces of the three disks as follows.

<u>File T</u> ools Preferences <u>H</u> elp								
📑 🔚 🏊 🧊 🗃			Ru	IN SYRT	THES 🕒 S	stop SY	RTHES 🔯 Calcula	tion Progress
Home		••)	-			T		
File Names	неа	texchange	Flux condit	ion D	irichlet condi	tion	Contact resistance	Infinite rac
Initial conditions	Heat	t exchange co	efficient (W/	m²/Deg	C)			
Boundary conditions		Type	External T	Coef h	References		User commen	ts 🔺
Physical properties Volumetric conditions	V	Constant 👻	90	1000	258	Exter	n faces of the disks	
Periodicity	1	Constant 👻	1					
User C functions Control	7	Constant 👻	j					
Output	1	Constant 👻	1					
Running options	7	Constant 👻	j					
	V	Constant 👻						
	V	Constant 👻						
	7	Constant 👻						
	V	Constant 👻						
		Constant 👻						
	4	Constant -	1					▼

Figure III.7: Boundary conditions Heat exchange

• Step 5: Define physical properties for each disk

Home File Names Isotropic Orthotropic Anisotropic Conduction Initial conditions Boundary conditions ρ (kg/m ³), Cp (J/kg/Deg C), k : Isotropic conductivity (W/m/Deg C) Physical properties γ Constant \checkmark ρ Cp k References User comments Volumetric conditions Periodicity γ Constant \checkmark γ Constant \checkmark Isotropic conductivity for disk 1 a User C functions Control γ Constant \checkmark γ γ γ	•••
Initial conditions Boundary conditions ρ (kg/m³), Cp (J/kg/Deg C), k : Isotropic conductivity (W/m/Deg C) Type ρ Cp k References User comments Physical properties Volumetric conditions Volumetric conditions Volumetric conditions Volumetric conductivity for disk 1 a Vser C functions Control Constant v 7700 460 25 1.4 Isotropic conductivity for disk 1 a Ver C functions Control Constant v Image: Constant v Image: Constant v Image: Constant v Image: Constant v	••••
Boundary conditions γ Type ρ Cp k References User comments Physical properties V Constant * 7700 460 25 1 4 Isotropic conductivity for disk 1 a Volumetric conditions V Constant * V V V V Verify Constant * V V V V V Volumetric conductive V V V V V	••••
Volumetric conditions Volumetric conductivity for disk 1 a Periodicity V User C functions V Constant • V Volumetric conductivity for disk 1 a	
Periodicity Image: Constant Imag	
Control	
Output	- 1
Running options	
Constant 👻	
✓ Constant ▼	
✓ Constant ▼	
Constant 👻	
Constant 👻	
Constant 👻	
4 Constant -	•

Physical properties Define Isotropic, Orthotropic and Anisotropic properties as follows.

Figure III.8: Isotropic properties on disks 1 and 4

Define the physical properties for the disk 2 with orthotropic conductivity.

🗋 🦢 ⊵ 🌆						Run S	SYRTHE	is 🕑 Stop	SYRTHES 🔞 Calculation Progres
Home									
File Names	Isotr	opic	Ortho	ropic	Aniso	tropic			
Initial conditions	0 (ku	ŋ/m³).	Cp (I/ka	(Deg C).	kx kv	Orthot	ropic co	onductivity (W	(m/Deg C)
Boundary conditions	P Vite		-r ()		Cn.	- Lov	la.	Boforoncoc	lleer commente
Physical properties		0	iype	7700	cp		- ×y	References	oser comments
Volumetric conditions	V	Cons	tant	//00	460	25	5	2	Orthotropic conductivity for disk 2
Periodicity	1	Cons	tant	-					
Control	1	Cons	tant	-					
Output	1	Conc	tant						
Running options		Cons	tant						
	1	Cons	tant	-					
	1	Cons	tant	-					
	1	Cons	tant	-					
	V	Cons	tant	-					
	J	Cons	tant	-					
	V	Cons	tant	-					
	V	Cons	tant	-					
	V	Cons	tant	-					
	V	Cons	tant	-					
	4	h							

Figure III.9: Orthotropic properties on disk 2

• **Remark**: To correctly identify the volume references associated to a specific physical property, we can check the mesh regions directly inside ParaVis.

Define the Physical properties for the disk 3 with anisotropic conductivity.

<u>File Tools Preferences Help</u>											
📑 🖆 🏊 🃭 🔊						Run S	YRTHE	🛚 🕒 Stop SYF	RTHES 🔕 (Calculation Progress	M
Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options	Isot φ ψ	ropic Orth g/m³). Cp (J/ Constant Constant Constant Constant Constant Constant Constant Constant Constant Constant Constant Constant	notro <pre></pre> <pre></pre>	popic peg C), ρ 7700	Aniso kx ky Cp 460	kun s tropic : Anisoti 25	sopic co	Angle (in Deg)	Deg C) References 3	User comments Anisotropic con	
	4	Constant	•							•	•

Figure III.10: Anisotropic properties on disk 3 Mesh

1.6 Control Tab

Define the global number of time steps and the time step for the 2D solid conduction computation.

Image: Conduction Progress Image: Conduction Progress Home File Names Conduction Initial conditions Boundary conditions File Stop Synthes Volumetric conditions File Stop Synthes Volumetric conditions File Stop Synthes Control Output Running options Image: Control	<u>File Tools Preferences Help</u>	
Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options Time management Restart management Solver information Time step management Global number of time steps : 100 Time step : Constant \neg Constant time step Time step (in seconds) : 10	📑 🗁 ⊵ 📭 🔊	Run SYRTHES 🕟 Stop SYRTHES 🛛 Calculation Progress 🧾
	Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options	Time management Restart management Solver information Time step management Global number of time steps : 100 Time step : Constant Ime step Constant time step Time step (in seconds) : 10

Figure III.11: Time management settings

1.7 Output

Define the probe coordinates for output management.

<u>File T</u> ools Preferences <u>H</u> elp		
📄 🖆 ⊵ 📭 📾	Run SYRTHES 🕟 Stop SYRTHES 🔇 Calculation Progress	M
Home File Names Conduction Initial conditions Boundary conditions Physical properties	Probes Result fields Surface balance Volume balance Frequency of output Every n time steps 1 Definition by coordinates	
Volumetric conditions Periodicity	X Y User comments	-
User C functions Control Output Running options	1 V 0.003 -1.2 disk1 2 V 3 V 4 V 5 V 6 V 7 V 8 V 9 V 10 V 11 V 13 V 15 V	-

Figure III.12: Probes creation

Define the frequency at which the results fields are written

<u>File Tools Preferences Help</u>	
📑 🚰 🏝 🌗 📾	Run SYRTHES 🕑 Stop SYRTHES 🚳 Calculation Progress 🗾
Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options	Probes Result fields Surface balance Volume balance Frequency at which the result fields are written in the transient result file (extension ".rdt") : 2 2 D Fields Every n time steps 25 2 D heat flux field 2 D maximum temperature field Disable final 2D fields Starting time (s) : Ending time (s) :

Figure III.13: Results fields window

EDF R&D

.

1.8 Running options

• Step 6: Running SYRTHES computation alone.

Listing file browser Listing file editor Log

S Y R T H E S SYRTHES2ENSIGHT : FIN NORMALE

Define the file name of the SYRTHES listing and the number of processors used.

Image:	<u>F</u> ile	<u>T</u> ools Preferences <u>H</u> elp		
Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options Control Output Running options Convert result for softwares : Ensight/Paraview Note Preprocessing for OD/LD fluid mesh Convert result for softwares : Ensight/Paraview Note Preprocessing for Softwares : Ensight/Paraview Note Preprocessing for Softwares : Ensight/Paraview Note Preprocessing for Softwares : Ensight/Paraview	:	🖆 📐 🚺 🗃	Run SYRTHES 🕟 Stop SYRTHES 🔞 Calculation F	rogress 📈
		Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity User C functions Control Output Running options	Scalar/ Parallel calculation : number of processor used for conduction : Scalar/ Parallel calculation : number of processor used for radiation : Listing name: listing Advanced options Preprocessing : automatic preprocessing for 0D/1D fluid mesh • Domain partitioning : automatic mesh partitioning using SCOTCH • Convert result for softwares : Ensight/Paraview •	1 () 1 () () () () () () () () () () () () () (

Figure III.14: Running options window

- Progress of Syrthes run	40.40
100%	37.86
Reset Scale	35.31
~Axes options yleft x yright	ີບູ 9 32.76
Nb. decimals: 2 2 2 2 2	30.21
Scientific format:	Č 27.66
Time scaling: second	F 25.12
G1 G2 G3 G4 G5 G6 G7 G8	22.57
History 🔹 Temp 👻	20.02
Line Style :	10.00 208.00 406.00 604.00 802.00 1000.00 Time (second)
● yleft ◯ hide ◯ yright	Temp disk1

The following graph represents monitoring point the temperature value versus time during computation

Figure III.15: Screenshot of the computation progress window

1.9 Preparing and launching the code_saturne computation alone

The main steps of the preparation of the fluid computation alone can be the following ones:

• Step 1: Launch code_saturne GUI from the object browser or the tool bar in SALOME module CFDSTUDY (or ./SaturneGUI in command line),

- Step 2: Create a New case,
- Step 3: Check mesh quality by running a Mesh quality criteria calculation,
- Step 4: Define Mesh settings and Boundary zones
- Step 5: Define Calculation features
- Step 6: Define Volume conditions
- Step 7: Define Boundary conditions
- Step 8: Define the Time Settings
- Step 9: Define postprocessing
- Step 10: Run the code_saturne computation alone.
- Step 1 & 2 : Create code_saturne New case and launch code_saturne

\$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
\$ code_saturne gui &

1.10 Mesh Tab

• Step 3: Check mesh quality by running a Mesh quality criteria calculation. Choose the fluid mesh with code_saturne (GUI) and select the execution mode *Mesh quality criteria only*. Click on run computation to check mesh quality.

Eile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 💼 🏝 😏 🕐 🔳 🖗	e 2 o 2 💶 🌣	
Ø 8		
Calculation environment	Mesh input	
Preprocessing	Import meshes Use existing mesh input Generate cartesian mesh	
Volume zones	Local mesh directory (optional)	
Boundary conditions		
Postprocessing		
Performance settings	List of meshes	
	3rond2d fluide des Simail/NOPO	
	* -	
	Execution mode	
	Mesh quality criteria only	
	✓ Use unmodified checkpoint mesh in case of restart	
	✓ Save mesh if modified by preprocessing	
Computation Advanced	t	
Corint parameters		
Script parameters		
	Result subdirectory name	
	Number of processes	
	Threads per process 1	
omputation start		
	Consel Apply Courses and an	a an I an I an I an I an I
	Cancel Apply Save case and run	n calculation

Figure III.16: Mesh quality criteria only computation

The generated *preprocessor.log* file, located in \boxdot **RESU** will be used in the next step.

• Step 4: Define Mesh settings and Boundary zones

Select the option *Standard computation* as execution mode and set up physical properties as follows. Load the preprocessor.log file inside the code_saturne (GUI) to define boundary regions.

Standard Computation	•
✔ Use unmodified checkpoint mesh i	n case of restart
V Save mesh if modified by preproce	ssing

Figure III.17: Standard computation execution option

Cancel	Select a preprocessor log	٩	Open
⊘ Récents	 ✓ FLUID RESU 20211001-0946 	Þ	
Dossier personnel	Nom	▼ Taille	Modifié
🖿 Bureau	 performance.log postprocessing 	2,8 ko	09:46 09:46
Documents	📄 preprocessor.log	5,8 ko	09:46
🖸 Images	run_solver.log	11,7 ko	09:46
J Musique			
Téléchargements			
▶ Vidéos			
🖿 DATA			
		Preprocesso	r log 🔻

Figure III.18: Preprocessor.log fil loading window

Boundary zones Once the boundary regions automatically loaded , define the boundary conditions as follows

<u>File Edit Tools W</u> indow <u>H</u> elp	
📑 눹 🖄 😒 🔳 🛛	2 6 2 6 2 🖪 🕸
C	8
Calculation environment	Boundary regions definition
👻 🕅 Mesh	
Preprocessing	Label Zone Selection criteria
Volume zones	BC_1 1 1
Boundary zones	BC_2 2 2 2
▶ An Calculation features	BC_3 3 3
 Wolume conditions 	
all_cells	
Zone_1	
 Boundary conditions 	
BC 1	
BC_2	
BC_3	
Coupling parameters	
$\rightarrow \Delta t$ Time settings	Add Delete
→ ∆x Numerical parameters	Add from preprocessor log
Postprocessing	Add from preprocessor log
Performance settings	Import groups and references from preprocessor log

Figure III.19: Loaded Boundary zones

Rename selection criteria and delete the third zone.

Label	Zone	e Selection criteria	
Wall	1	1	
Symmetry	2	2 or 3	
		Add Delete	
dd from pr	eproce	Add Delete	
dd from pr	eproce	Add Delete	

Figure III.20: Boundary zones updated

EDF R&D

1.11 Calculation features Tab

• Step 5: Define Calculation features

Select Incompressible Standard Eulerian single phase Flow Models as follows

ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
📑 눹 🖄 🔅 🔳 🖗	6 🛛 6 2 🖪 🌣		
	Flow Models Flow Models Standard Eulerian single phase Atmospheric Electric arcs Groundwater Reactive flows (combustion) Homogeneous Eulerian - VoF mode Eulerian multiphase (neptune_cfd)	Incompressible	•
Performance settings	Additional Features Particles and droplets tracking Turbomachinery model Deformable mesh (ALE method) Fans (source-term model)	off None	•

Figure III.21: Calculation features window

Turbulence models Select the $k - \varepsilon$ LP as turbulence model

<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp	
📄 눹 🖄 🕱 🥐 🔳 🖗	6262
2 B	
Calculation environment	Turbulence model
▶ 🕅 Mesh	
▼ Av Calculation features	k-ε Linear Production 👻
Turbulence models	
🖹 Thermal model	
Body forces	Advanced options 🐰
Species transport	
 Wolume conditions 	Peterence values (used for initialization of turbulance)
Boundary conditions	Reference values (used for initialization of carbutence)
	Velocity scale 1.0 m/s
▶ ∆x Numerical parameters	
Postprocessing	Length scale Automatic 👻
Performance settings	

Figure III.22: Turbulence models

Thermal model Select Temperature (Celsius) under Thermal Scalar header

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🗎 🔔 🥱 🖻 💀	6262
0 10	
Calculation environment	Thermal scalar
🕨 🕅 Mesh	
▼ div Calculation features	Temperature (Celsius)
Turbulence models	The second se
📄 Thermal model	i nermal radiative transfers
Body forces	No radiative transfers
🚡 Species transport	
Volume conditions	
Boundary conditions	
Coupling parameters	
▶ ∆t Time settings	
▶ ∆x Numerical parameters	
 Postprocessing 	
Performance settings	

Figure III.23: Thermal Scalar

Body forces Add 1 G acceleration value following y axis as follows

ne Eair Tools window Help		
📄 💼 📐 👌 🕐 🔳 👳		
0 8		
Calculation environment	Gravity	
▶ 🕅 Mesh		
 ••• ••• Calculation features 	g _x 0.0 m/s ²	
Turbulence models	a9.81 m/s ²	
Thermal model	5y	
🔋 Body forces	g _z 0.0 m/s ²	
Species transport		
Volume conditions	Coriolis source terms (rotation vector)	
Boundary conditions		
Coupling parameters	Ω _x 0.0 s ⁻	
→ ∆t Time settings	$\Omega_{\rm v}$ 0.0 s ⁻¹	
▶ ∆x Numerical parameters		
Postprocessing	$\Omega_z = 0.0$ s ⁻²	
Performance settings		

Figure III.24: Gravity settings

1.12 Volume conditions

• Step 6: Define Volume conditions

Tick Initialization and Physical properties



Figure III.25: Volume zone types

Initialization Initialize temperature value to 20°C

	FLUID : setup.xml - code_saturne	×
<u>File Edit Tools Window H</u> elp		
📄 🖆 🧆 🤌 🖉 🔳 🖗 🙆	2 🖪 🌣	
08		
Calculation environment Physical	properties Initialization	
Calculation features	Inaramatars	
✓ The Volume conditions	Mathematical expression editor X	
all_cells	mathematical expression curtor	
Boundary conditions User expres	ssion Predefined symbols Examples	
Coupling parameters		
→ Δx Numerical parameters	ure = 20.;	
▶		
Performance settings		
	● Cancel	

Figure III.26: Initialization of the temperature.

📑 🛅 😒 🥱 🙋 🔜 🖗	
8	
Calculation environment Calculation features	Physical properties Initialization
Wolume conditions all_cells Boundary conditions Boundary conditions Coupling parameters ∆r Time settings	Material user_material ~ Method user_properties ~
Δx Numerical parameters	Reference total pressure
Postprocessing Performance settings	value 101325.0 Pa
	Reference temperature
	value 20.0 ° C
	(used for properties initialization)
	Density
	user law • 22 Reference value p 1.17862 kg/m ³
	Viscosity
	constant Reference value µ 1.83e-05 Pa.s
	Specific heat
	constant •
	Reference value C_p 1017.24 J/kg/K
	Thermal conductivity
	constant *
	Reference value λ 0.02495 W/m/K

Physical properties Define all physical properties as follows

Figure III.27: Define the physical properties.

User expression	Predefined symbols	Examples	
density = p0/	(287.0*(temperatu	re+273.15));	
		Cancel	∉ок

Figure III.28: Define the variable density with an ideal gas law.

1.13 Boundary conditions

• Step 7: Define Boundary conditions

First, check boundary zone types. Parameters should be set as follows : Define a thermal transfer

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 🖹 😏 党 🔳 🖗 🚺	o 🛛 🖸	0
Calculation environment Calculation environment Calculation features Calculation fea	Boundary zon Label Walt Symmetry	Nature Wall Symmetry
1		

Figure III.29: Boundary conditions - Zone types

condition as wall boundary condition with an external wall temperature $T_{ext} = 30^{\circ}$ Cand an exchange coefficient $q_{ext} = 10 \ (W/m^2.K)$.

	38	
Calculation environment Calculation features Calculation features Volume conditions	Wall [Wall] Smooth or rough wall	
Boundary conditions Wall	Smooth wall O Rough wall	
 Symmetry Coupling parameters Δt Time settings 	Sliding wall	
Δx Numerical parameters Postprocessing	Thermal	
泰 Performance settings	Type Exchange coefficient * temperature * Value 30.0 Exchange coefficient 10.0	

Figure III.30: Boundary conditions - Wall

1.14 Time settings Tab

• Step 8: Define the Time Settings

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📑 🖆 🏝 🥱 🥐 🔳 🖗 🌘	
Image: Second	Time step option Constant Velocity-Pressure algorithm SIMPLEC
	Reference time step 0.1 s Limitation by local thermal time step
 Postprocessing 	Stopping criterion Number of time steps

Figure III.31: Time settings windows

1.15 Postprocessing Tab

• Step 9: Define postprocessing

Define the writer and frequency output inside the code_saturne (GUI) as follows.

<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp					
📑 🗎 🕭 🤌 🔳 🖗	i 🗧 🖉 🧕 🔁 🚺 🔅	E			
0 🛙					
Calculation environment	Output Control Writer	Mesh Monito	rina Points		
Mesh					
▶ Ø Calculation features	Name	Id		Format	Directory
 Wolume conditions 	results	-1		EnSight	postprocessing
Image: Boundary conditions					
Coupling parameters					
▶ ∆t Time settings					
▶ ∆x Numerical parameters					
Postprocessing					
Additional user arrays					
Time averages					
Volume solution control					
Surface solution control					
Profiles					
Balance by zone					
Performance settings			+ -		
	F				
	Frequency				
	Output every 'r	n' time steps	-	25	
				_	
	Output at sta	rt of calculation		 Output at en 	d of calculation
	Time-dependency				
		Fixed r	nesh	•	
	Options				
		✓ Separate s	ub-writer for	each mesh	
		File type	binary (nati	ve) –	
		Polygons	display	-	
		Polyhedra	display	•	
4					

1.16 Run computation

• Step 10: Run the code_saturne computation alone. Launch the calculation by clickin on the

Computation	Advanced				
Script paramet	ers				
		Result subdirectory name Number of processes Threads per process	1	4	
omputation sta	ırt				

Figure III.33: Run computation

1.17 Postprocessing Analyses



Figure III.34: Visualization of the 2D fluid velocity field



Figure III.35: Visualization of the 2D fluid temperature field

1.18 Preparing and launching code_saturne-SYRTHES coupled computation

In order to create a coupled compution between SYRTHES and code_saturne some settings must be applied.

The last modification to prepare the coupling computation are given below:

- Step 1: Activate conjugate heat transfer in SYRTHES GUI,
- Step 2: Activate conjugate heat transfer in code_saturne GUI,
- Step 3: Give identical number of iterations and reference time step for both codes,
- Step 4: Check the run.cfg python script and launch the calculation by executing the runcase.

You will, first, set SYRTHES parameters then code_saturne parameters.

• Step 1: Activate conjugate heat transfer in SYRTHES GUI

1.19 Coupling SYRTHES- Syrthes coupling

<u>File T</u> ools Preferences <u>H</u> elp		
📑 🗁 🏝 🌗 🔊	Run SYRTHES (>) Stop SYRTHES	🔇 Calculation Progress 📈
Home File Names Conduction Conjugate heat transfer User C functions Control Output Running options	Case title : 3_2D_DISKS_coupled User description of the case	v 5.0

Figure III.36: SYRTHES Home Window - Conjugate Heat Transfer option ticked

Specify the references of the boundary zones for the coupling surfaces with code_saturne.

1.20 Coupling SYRTHES- Conjugate heat transfer Tab

<u>F</u> ile <u>T</u> ools Preferences <u>H</u> elp	
📑 🚰 🏝 🣭 🔊	Run SYRTHES 🕟 Stop SYRTHES 🔞 Calculation Progress 🗾
Home File Names Conjugate heat transfer User C functions Control Output Running options	Surface coupling Volumetric coupling Definition of the conjugate heat transfer coupling with CFD codes. Image: strategy of the CFD code instance Image: strategy of the CFD cod

Figure III.37: Conjugate heat transfer tab - Surface coupling

1.21 Coupling SYRTHES- Control Tab

Change the number of iterations and reference time step for the solid domain.

Image: Control Contrel Control Control Control Control Control	<u>File T</u> ools Preferences <u>H</u> elp	
Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity Conjugate heat transfer User C functions Control Output Running options	📑 🖆 🏝 🃭 🔊	Run SYRTHES 🕟 Stop SYRTHES 🔕 Calculation Progress 🗾
	Home File Names Conduction Initial conditions Boundary conditions Physical properties Volumetric conditions Periodicity Conjugate heat transfer User C functions Control Output Running options	Time management Restart management Solver information Time step management Global number of time steps : 600 Time step : Constant * Constant time step Time step (in seconds) : 0.5

Figure III.38: Control Tab - Time Management

This step closes SYRTHES settings. Now you need to set code_saturne parameters.

• Step 2: Activate conjugate heat transfer in code_saturne GUI,

1.22 Coupling code_saturne- Boundary conditions Tab

First, you must activate **SYRTHES coupling** under **Boundary conditions** and fill SYRTHES instance field. The latest must be the name of the SYRTHES case previously created. In our turorial the case name is : **SOLID**

File Edit Tools Window Help			
📄 🛅 🖄 🥱 🦿 📕 😔 🚺			
	08		
Calculation environment	Wall [Wall]		
Mesh			
▶ div Calculation features	Smooth or rough wall		
 Wolume conditions 	Creativelly C Development		
 E Boundary conditions 	• Smooth wall O Rough wall		
📄 Wall			
Symmetry	Sliding wall		
Coupling parameters			
→ ∆t Time settings			
 	Thermal		
 Postprocessing 	inemat		
Performance settings	Type SYRTHES coupling 🔹 🕅		
	temperature 💌		
	SVDTHES instance COUD		
	STRIES instance SOLID		

Figure III.39: Activate Thermal SYRTHES Coupling

1.23 Coupling code_saturne- Coupling parameters Tab

Set SYRTHES parameters as follows :

<u>File Edit Tools Window H</u> elp				
📄 💼 🏝 🕱 👌 💌 🔯	202	a 🔅		
2 8				
Calculation environment Mesh	Internal coupling	Syrthes coupling	Fluid structure interaction	
Calculation features We Colume conditions	Verbesitulous	0		
E Boundary conditions Coupling parameters	2D annia than ania	0		
 → Δt Time settings → Δx Numerical parameters 	2D projection axis	Z		
Exprocessing Performance settings	Relative tolerance	0.1		
	SYRTHES instances			
	Instance name		Boundary zones	
	SOLID		BC_1	

Figure III.40: code_saturne SYRTHES Coupling parameters

• Step 3: Give identical number of iterations and reference time step for both codes,

1.24 Coupling code_saturne- Time settings Tab

Change the number of iterations and reference time step for the fluid computation. It is important to set the same values as SYRTHES time settings.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
0	8			
Calculation environment				
🕨 🕅 Mesh	The start when			
 <i>w</i> Calculation features 	lime step option Constant			
Volume conditions	Velocity-Pressure algorithm SIMPLEC 🔹			
Boundary conditions				
Coupling parameters				
 M Time settings 				
Start/Restart	Reference time step 0.5 s			
▶ ∆x Numerical parameters	Limitation by local thermal time step			
 Postprocessing 				
Performance settings				
	Stopping criterion Number of time steps			

Figure III.41: Equalizing time settings parameters

1.25 Coupling code_saturne- Numerical Parameters Tab

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp					
ମ ସ					
Calculation environment					
Mesh Galculation features	Gradient reconstruction type Green-Gauss with iterative handling of non-orthogonalities 💌				
We would be conditions We Boundary conditions					
Coupling parameters	✓ Handling of transposed gradient and divergence source terms in momentum equation				
→ Δt Time settings					
 Mumerical parameters 					
Equation parameters					
 Postprocessing 	Pseudo-coupled velocity-pressure solver				
Performance settings	✓ Improved pressure interpolation				
	Relaxation of pressure increase 1.0				
	Algorithm to take into account the density variation in time Automatic				

Activate the Improved pressure interpolation.

Figure III.42: Numerical parameters updated

• Step 4: Check the run.cfg python script and launch the calculation by executing the runcase.

1.26 Coupling Running computation

• Substep 4.1 - Check/Edit run.cfg file settings: After having enabled conjugate heat transfer on both sides (in the fluid and in the solid data sets), having set equal time step parameters on both sides and having checked/edited (if needed) the run.cfg script as follows:

>[setup] >coupled_domains: FLUID:SOLID >n_procs: >n_threads: >[fluid] >domain: FLUID >n_procs_weight: 1 >solver: Code_Saturne >[solid] >domain: SOLID >n_procs_weight: 1 >opt : -v ens >param: 3_2D_DISKS_coupled.syd >solver: SYRTHES

you can process to the launch of computation.

Substep 4.2 - Run the case: Verify the run.cfg file is well filled. names and param: should be updated.

Finally, launch the **runcase** present in the study directory (named in our case $\boxdot 3_2D_JISKS$) and run the coupling computation via a terminal command as follows:

\$ code_saturne run

• Note:: in the run.cfg, the number of processors can be specified for each code (as this example with 1 processors for code_saturne and 1 processors for SYRTHES). It can be either both codes in parallel, one in parallel and the other one in sequential, or both in sequential.

One can specify the ouput results format for SYRTHES with an option (opt) which takes the value -v ens for a 3D fields output with a EnSight format or -v med for a 3D fields output with a SALOME format).